



STATIC ANALYSIS OF A 6 - AXIS INDUSTRIAL ROBOT USING FINITE ELEMENT ANALYSIS

Supriya Sahu and B.B. Choudhury

Department of Mechanical Engineering,

Indira Gandhi Institute of Technology, Sarang, Odisha, India

(An Autonomous Institute of Govt. of Odisha)

ABSTRACT

The present work aims to find out the Deformation, Stresses, Shear Elastic Strain and Strain Energy at different points of an industrial robot to determine it's the safety factor by using Finite Element Method (FEM). Six axis industrial robots are generally used in industries for various production works such as pick and place and for different operations. So it needs to be properly designed. The model of robot is established using the ANSYS software and finite element analysis is done. Different values of typical gripper loads are applied, and values at different conditions are compared to find out the weak parts, so further design improvement can be done.

Key words: ANSYS, deformation, finite Element, industrial robot and shear stress.

Cite this Article: Supriya Sahu and B.B. Choudhury, Static Analysis of a 6-Axis Industrial Robot Using Finite Element Analysis. *International Journal of Mechanical Engineering and Technology*, 8(3), 2017, pp. 49–55.

<http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=3>

1. INTRODUCTION

Six axis industrial robots are manufactured to industrial standards for training, research and industrial purposes. It is used for various purposes in industries for various works. The main objective of structural design is to produce structures capable of resisting applied load without failure. Improper designed of fabricated structures will fail if the applied load exceeds the design specification which may cause various problems in industries. So a well-designed structure will greatly minimize the cost of failure.

Deformation and Stress analysis are the primary task in designing the structure such as industrial robots. These are needed for the maintenance and investigation of such structural failure. There are many methods to determine stress in structures subjected to various loads applied. It is required to investigate the stability, strength and rigidity of structures.

(Chitte et al.) analyzed the weight of Six axis ARISTO robot structure for minimizing its structural deformations on directions Ox, Oy, Oz in order to improve the stiffness, on the basis of calculus of the loading forces applied in a static study. (Gasparetto et al.) analyzed a method and modeling has been done on industrial robots of low weight based on rigid link system approach. Based on the finite element method (Ghiorghe,) determined the optimum values for the design parameters considering the criteria of reducing the material used to build

the robot.(Hardeman et al.,) used finite element method to derive dynamic equations of motions that suitable to simulation and identification. (Ying Huang et al.,) analog simulation experiment is conducted using finite element analysis. (Jeevan and Rao) is performed for analysis the structural parameters and found that the circular shaped sustains more vibrations than square shaped robot arm structures. . (Kumar et al.) analyzed femur bone to get natural vibration, frequencies, modes and to know the fracture location in the bone through simulation using FEA and they compared the results with experimental results. (Kumar and Sambaiah,) they analyzed the robot to put an object in right location with correct amount of force and torque at right time using FEA. (Nor et al.) performed stress analysis ,modeled and simulated an actual low loader structure and determined maximum deflection and high stress value by the use of FEA. (Pachaiyappan et al.) analyzed the articulated robot arm taking into account the various critical loads acting on the base arm using a commercially available analysis tool, ANSYS. (Ristea,) analyzed the differences between composite substance and aluminum for designing the robot. The control system for a motorized robot arm has been designed. (Zhang and Cai,) proposed a method of compensation based on the error model with the 6-axis robot's parameters of kinematic structure and the joint angle and found that the main factor of the dynamic error is the deformation of the connecting rods. (Wang and Chen) investigated the mechanical casting defect of entrapped air bubble against static and fatigue strength by practical experiment and FEM model.(Yadav et al.) analyzed the robot pedestal using ANSYS and found that the natural frequency of the pedestal is minimum at first mode and maximum at sixth mode.

2. EXPERIMENTAL ANALYSIS

The different parts or members of 6 axis industrial robots are base, link, shoulder, elbow, wrist, Pitch and roll is shown in figure1.Each and every member are having different range of motions.

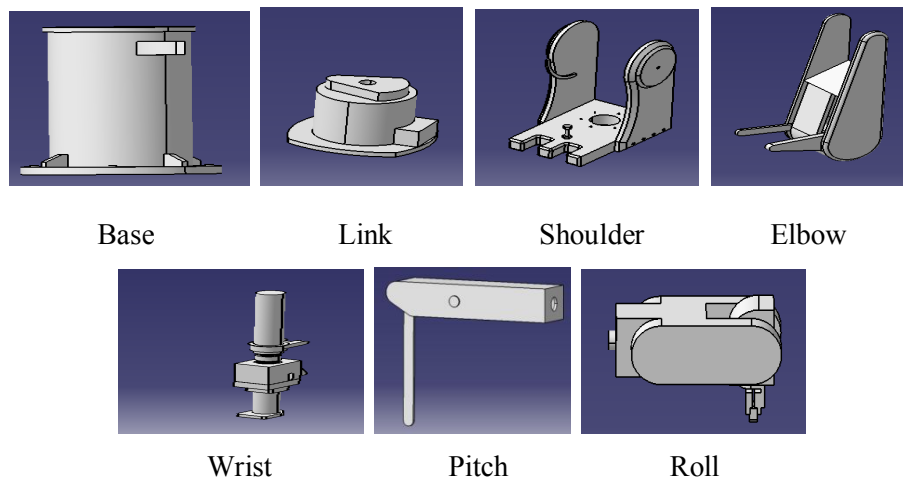


Figure 1 CATIA Models of the members of 6 axis industrial robot

Base is also called as waist. Being the first axis can revolve from 90° to 250° . It is the lowest or supporting part or structure of the robot. Shoulder is the second axis, it can play between -45° & -90° which will help the robot to moves in forward and backward. Elbow is the third axis, plays between 45° & 115° providing up and down movement for picking the object. Wrist is the fourth axis which can perform rotation from 0° to 340° . An angle of -90° to 90° can be obtained from this fifth axis called pitch which makes the robot flexible enough to pick 'n place the objects. Roll is the small additional wrist in place of sixth axis will revolve

from 0° to 340° , which gives much more flexibility to handle job-oriented objects. The robot assembly consists of members or parts is shown in figure 2.

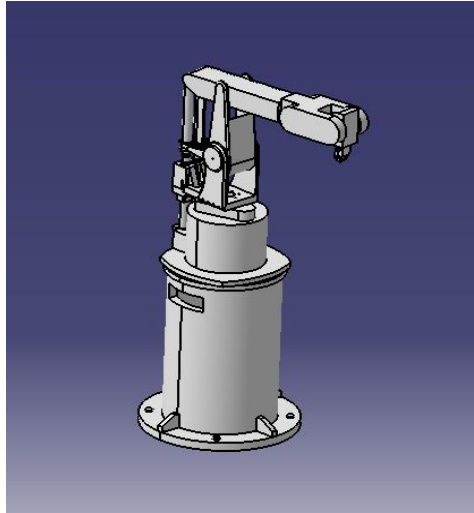


Figure 2 The model of robot assembly.

2.1. The Finite Element Model Analysis by ANSYS

First of all the model geometry of robot manipulator is prepared using CATIA V5 software which is used to establish three dimensional model. Then it is imported from CATIA to ANSYS workbench for FE analysis. Through the software interface the data exchange of the model is imported to the ANSYS. Structural steel is taken as the material of robot body as this type of six axis industrial robot is to be fixed on the floor while doing different operations. All nodes at the base were constrained in all directions. Proper material, weight factor and mesh size are taken for the FE modeling and analysis and loads are applied to the gripper. The mesh size is taken as 0.01.

In this analysis eight different gripper loads are considered starting from 0.5 N to 125 N, which are applied to the end effectors of gripper. The structure is tested to a static analysis in order to obtain total deformation, shear stress, shear strain and strain energy.

3. RESULT AND INTERPRETATIONS

The total deformation, maximum shear stress, maximum shear elastic strain and strain energy for eight different loads applied to the gripper are given in Table.1 below.

Table 1 Values of deformation, stress, strain and strain energy on different loading conditions

Sl no.	Gripper loads(N)	Total deformation (mm)	maximum Shear Stress(Pa)	Maximum Shear elastic Strain(m/m)	Strain Energy(J)
1	0.5	$5.2701e^{-7}$	42425	$5.5152e-007$	$4.1274e-010$
2	2.5	$2.635e^{-6}$	$2.1212e+005$	$2.7576e-006$	$1.0319e-008$
3	5	$5.270e^{-6}$	$4.2425e+005$	$5.5152e-006$	$4.1274e-008$
4	25	$2.635e^{-5}$	$2.1212e+006$	$2.7576e-005$	$1.0319e-006$
5	50	$5.2701e^{-5}$	$4.2425e+006$	$5.5152e-005$	$4.1274e-006$
6	75	$7.9052e^{-5}$	$6.3637e+006$	$8.2728e-005$	$9.2867e-006$
7	100	0.0001054	$8.485e+006$	$1.103e-004$	$1.651e-005$
8	125	0.00013175	$1.0606e+007$	$1.3788e-004$	$2.5796e-005$

Static analysis is important to assess the total deformation in order to know the safety factor of the structure. Total deformation of robot model structure for gripping load of 0.5 N and 125 N is shown in Figure 3. The dark blue colour indicates the lowest value of deformation, light blue colour shows lower value of deformation, yellow colour indicates higher value of deformation and red colour shows the highest value of deformation which is shown in the left side of the figure. From the figure it is seen that the lowest deformation is found at the base of robot and the highest deformation occurred near the gripper.

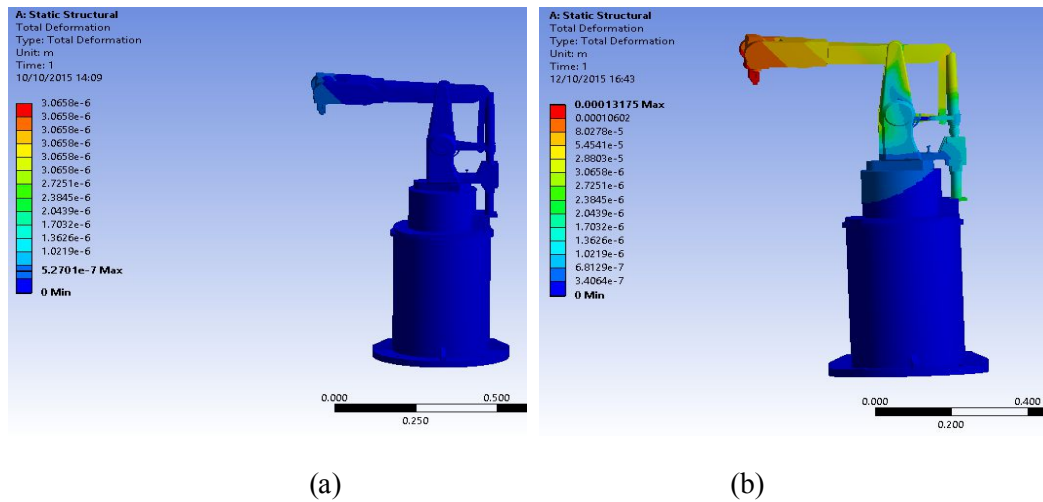


Figure 3 Total deformation of robot model structure (a) gripping load of 0.5 N (b) gripping load of 125 N

It is observed from the analysis that the deformation increases as the applied load to the gripper increases. From the different colour of the model it is seen that for force value of 0.5 N the model is totally blue in colour, that means it is the safest load that can be applied to the robot but for 125 N the maximum deformation is found which is seen as red colour near the gripper.

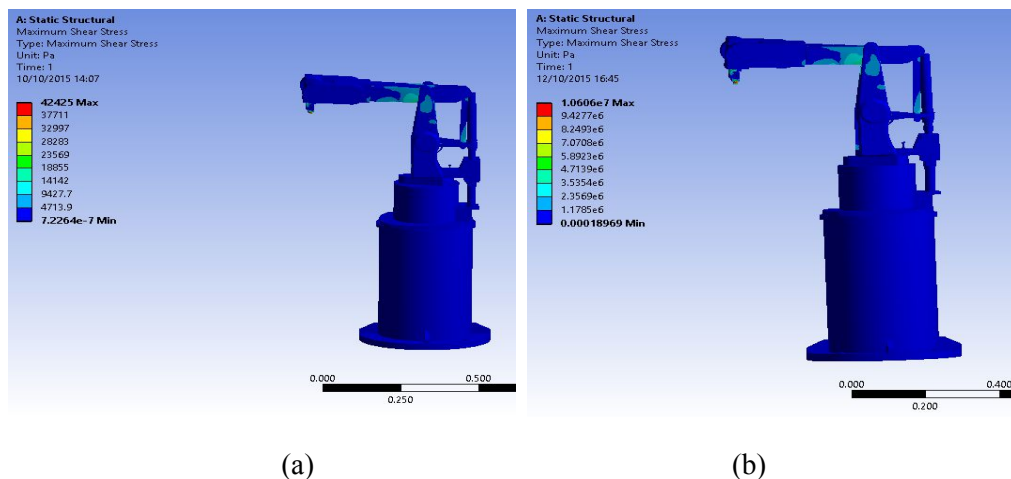


Figure 4 Shear Stress of robot model structure (a) gripping load of 0.5 N (b) gripping load of 125 N

Shear Stress of robot model structure for gripping load of 0.5 N and 125 N is shown in figure 4 and it is found that for load of 125 N the stress value is much more than for load of 0.5 N.

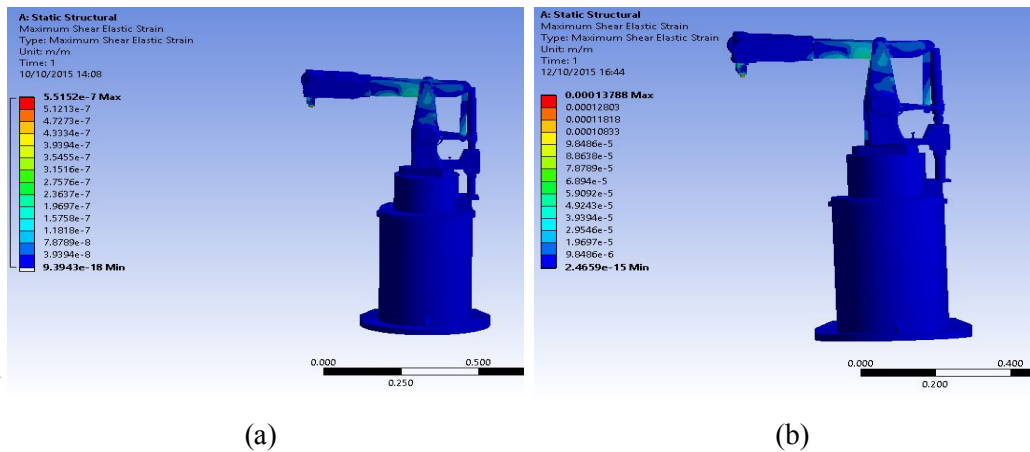


Figure 5 Shear Elastic Strain of robot model structure (a) gripping load of 0.5 N (b) gripping load of 125 N

It is seen in figure 5 that as the load increases the shear elastic strain value increases and the maximum value is shown in red colour and the minimum value is shown by blue colour in colour scale which is at the left side of the figure.

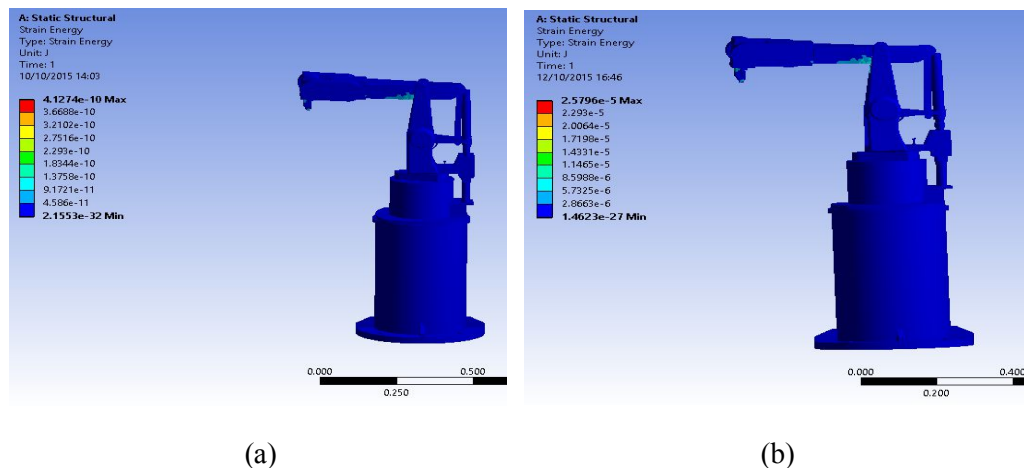
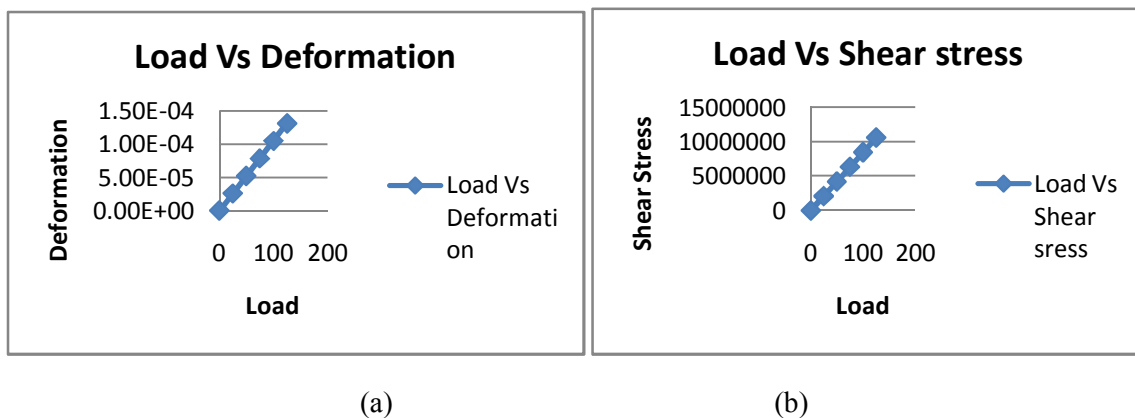


Figure 6 Strain Energy of robot model structure (a) gripping load of 0.5 N (b) gripping load of 125 N

Strain energy gives the idea of failure of a structure when strain energy per unit volume reaches the failure value of strain energy for that given structure. The strain energy values are shown in figure 6 for 0.5 N and 125 N force, where it is maximum for 125N force.



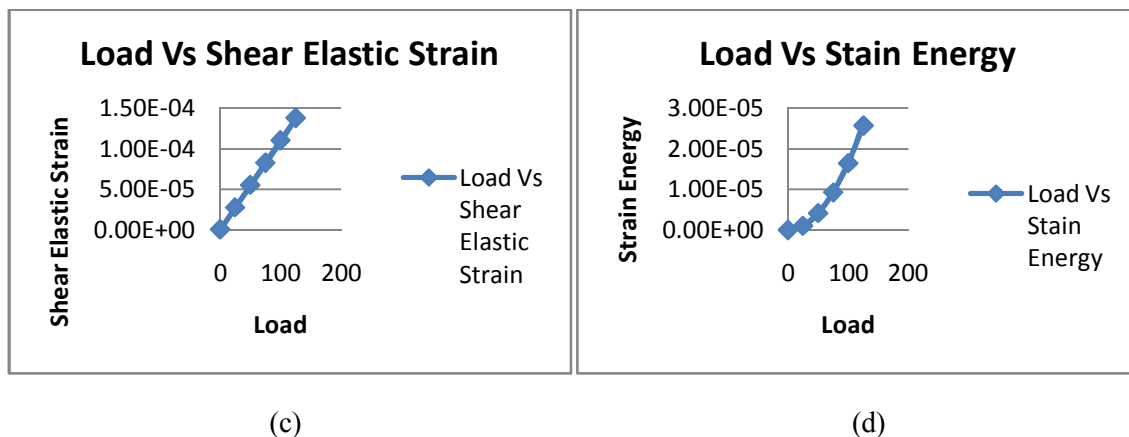


Figure 7 Graphical Presentation (a) Load Vs deformation (b) Load Vs Shear Stress (c) Load Vs Shear Elastic Strain (d) Load Vs Strain Energy

4. CONCLUSION

The current research aimed to reduce the weight of the structure and minimizing its structural deformations to improve the stiffness, on the basis of the loading forces applied in a static study. Some variants may be chosen from better shape, weight, stiffness, static and dynamic behavior.

The results obtained from the finite element analysis with mesh size of 0.01 are considerably good where investigation on maximum deformation, maximum stress and maximum strain is well done. The implementation of a FE based model is very much needed to simulate the behavior of an industrial robot. In this manner, from FEM the knowledge of maximum values of the factors which may lead to failure of the structure can be avoided in some later point in time.

REFERENCES

- [1] P. G. Chitte, Mr. S. S. Bansode, Mr. S. S. Rathod, Mr. N. S. Motgi, Structural and Vibrational Analysis of Six axis ARISTO robot using ANSYS, June 2016, IJIRT, Volume 3 Issue 1, ISSN: 2349-6002.
- [2] Alessandro Gasparetto, Amir Kiaeian Moosavil, Paolo Boscariol and Macro Giovagnon, Experimental validation of a Dynamic Model for Lightweight Robot, Int. J Adv Robotics System, Vol.10, 2013, 182:201.
- [3] Adrian ghiorghe, optimization design for the structure of an RRR type industrial robot, U.P.B. Sci. Bull., 2010, Series D, Vol. 72, Iss. 4,
- [4] Toon Hardeman, Ronald Aarts and Ben Jonke, Modelling and identification of robots with joint and drive flexibilities, IUTAM Symposium on Vibration control of Nonlinear Mechanisms and structures, 2005, 173-182.
- [5] Ying Huang, Panfeng Huang, Min Wang¹, Lei Wang, and Yunjian Ge, The Finite Element Analysis Based on ANSYS Pressure-Sensitive Conductive Rubber Three-Dimensional Tactile Sensor', CCIS 2, 2007, pp. 1195-1202.
- [6] Jeevan and Amar Nageswara Rao, Modeling and Analysis of Robot ARM using ANSYS, International Journal of Scientific Engineering and Technology Research, ISSN 2319-88, Vol.04, Issue.33, August-2015, Pages:6692-6697.
- [7] Ashwani Kumara, Himanshu jaiswala, Tarun Garga, Pravin P Patil, Free Vibration Modes Analysis of Femur Bone Fracture Using Varying Boundary Conditions Based on FEA, Procedia Materials Science, 2014, 1593 - 1599.

- [8] K.Manoj Kumar and Ch. Sambaiah, Modelling And Study of Motion Control System for Motorized Robot Arm Using Mat Lab And Analysis of The Arm By Using Ansys, International Journal of Mechanical and Industrial Engineering (IJMIE), 2012, ISSN No. 2231 –6477, Vol-2, Issue-1.
- [9] Mohd Azizi Muhammad Nor, Helmi Rashid, Wan Mohd Faizul Wan Mahyuddin, Mohd Azuan Mohd Azlan, Jamaluddin Mahmud, Stress Analysis of a Low Loader Chassis, Procedia Engineering , 2012, 995 – 1001.
- [10] S. Pachaiyappan, M. Micheal Balraj and T.Sridhar. Design And Analysis of an Articulated Robot Arm for Various Industrial Applications, Journal of Mechanical and Civil Engineering, 2014, e- ISSN: 2278-1684, p-ISSN : 2320–3334X, PP 42-53
- [11] Alin Ristea, FEA Analysis for Frequency Behavior of Industrial Robot's Mechanical Elements, Sinaia, Romania , 2011 , 26-28.
- [12] J.Zhang and J. Cai, Error Analysis and Compensation Method Of 6-axis Industrial, International journal on smart sensing and intelligent systems VOL.6, NO.4,september 2013
- [13] T.M. Wang and C.H. Chen, Fatigue Analysis of Wrist Joint Part of Six-Axis Articulated Robo, the 14th iftomm world congress, Taipei, Taiwan, 2015 October 25-30
- [14] G.AYadav, M.M Jadhav, V. Patil, Design Analysis and Optimization of Robot Pedestal, International Journal of Engineering and Technical Research (IJETR) ISSN: 2321-0869 (O) 2454-4698 (P), Volume-4, Issue-4, April 2016.
- [15] Kinan. Dandan, Anani. Ananiev, Ivan. Kalaykov. Dynamical Analysis of Silo Surface Cleaning Robot Using Finite Element Method, *International Journal of Mechanical Engineering and Technology*, 7 (1), 2016, pp. 190-202.
- [16] S. S. Deshpande, P. N. Desai, K. P. Pandey and N. P. Pangarkar, Consistent and Lumped Mass Matrices in Dynamics and their Impact on Finite Element Analysis Results. *International Journal of Mechanical Engineering and Technology*, 7(2), 2016, pp. 148–167.